Non Linear Finite Element Analysis on Confined Concrete Columns under Concentric and Eccentric Compression

Arathy Nath P[1], Indu Susan Raj[2]

[1]M.Tech Student, Depatment of Civil Engineering,IIET,Kerala, India

[2]Asst. Professor, Department of Civil Engineering,IIET,Kerala, India

Abstract: ***For a particular load, it is possible to use different diameter for longitudinal bars, which determines the number of bars and hence the arrangement of reinforcement. This paper studies a nonlinear approach of finite element analysis for twelve square reinforced concrete columns under concentric and eccentric compressive loadings. The diameter for longitudinal bars selected for this study includes 10mm, 12mm and 16mm. All the specimens were laterally reinforced with 6mm ties at 50mm spacing.The behaviour of different arrangement of reinforcement for a column for a particular load is analysed using ANSYS 14.5 finite element software and the results were compared.Results indicate that all the models shows a definite change in the load carrying capacity.***

Keywords: finite elements, arrangement of reinforcement, concentric and eccentric compression, ultimate load, load for first crack, maximum deformation

1. Introduction

Columns are vertical compression members of a structural frame intended to support the load-carrying beams. They are provided to tansmit loads from the upper floor to foundation below. The main cause of collapse of many structures such as failure of bridges, multistoreyed buildings is due to the failure of supporting members such as columns. The stability of column depends on many factors. Out of such factors, the design of reinforcement plays an important role in the stability of column. There are two types of reinforcement mainly provided: longitudinal and transverse reinforcement. Longitudinal reinforcement are main reinforcement serves to support the compression load coming on it. The main purpose of transverse reinforcement is to maintain the longitudinal reinforcement in position and avoid the tendency of deflection. In addition to that the insufficient amount of transverse reinforcement may affect the ductility of column. Lack of ductility is the ultimate reason for most of the failure of supporting members.

The main cause of collapse of structues are due to the lack of required amount of transverse reinforcement. Hence both longitudinal and transvere reinforcement plays a good role in the stability of column. Transverse steel act as confining steel for concrete core in column. It avoid the sudden collapse of columns. The amount, spacing and arrangement of transverse reinforcemnt depends on longitudinal reinfocement. Transverse reinforcement are provided in the form of lateral ties , circular rings or helical reinforcement. They are provided to ensure that every longitudinal bar nearest to the compression face has effective lateral support against buckling.

This paper studies a nonlinear approach of finite element analysis of twelve square reinforced concrete columns confined with transverse steel under concentric and eccentric compressive loading. For a particular load, we can use different diameter for longitudinal bars, which determines the number of bars and hence the arrangement of reinforcement. The behaviour of different reinforcement arrangement for column for a particular load is analysed using ANSYS 14.5 finite element software and the results were compared.

2. Analytical Works

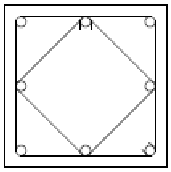
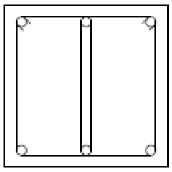
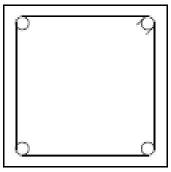
Finite element analysis (FEM) is a numerical method for the analysis of field problems the field problems in engineering and physics for example, the structural analysis problems, heat transfer problems, fluid flow problems etc. Whether in 1D 2D or 3D , the FEM can be used effectively. One of the reasons for wide applications of the FEM is due to the availability of a number of general purpose analysis programs. In FEM, a complex region defining a continuum is discretized in to small geometric shapes called elements. The properties and governing relationships are assumed over these elements and expressed mathematically in tems of unknown values at specific points in the elements called nodes. An assembly process is used to link the individual elements to the given system. When the effects of loads and boundary conditions are considered, a set of linear or non linear algebraic equations is usually obtained solution of these equations gives the approximate behaviou of the continuum or system While the continuum is having a infinite number of degrees of freedom the discretized model have finite degrees of freedom.

The whole analysis procedure is well organized in to three basic components: pre processing, processing and post processing. A preprocessor creates the finite element model and input data necessary for a finite element analysis. The post processor accepts the results of analysis and generates tables, diagrams, graphs etc. for interpretation of results. The preprocessor accepts input from the used created finite element mesh and other data required for analysis and displays the model for data check and correction ,if any , to be made by the user in an interactive mode. Graphical post processing of the results helps to visualize the physical consequences of the analysis.

2.1 Geometry Modeling

In this study, twelve columns were analysed. The columns have a typical cross section of 200 × 200mm with a height of 800mm. The cover provided to concrete is 15mm. Out of 12 column specimens, four column specimens were reinforced with 10 mm longitudinal bars, four with 12 mm longitudinal bars and remaining four with 16mm diameter longitudinal bars. Transverse reinforcement are provided in the form of lateral ties, circular rings or helical reinforcement. In this study, all the specimens were laterally reinforced with 6mm ties at 50mm spacing.

Table 1: Properties of models used

Specimen	Eccentricity (mm)	Reinforcement	Section
Model 1	0 30 60 90	Longitudinal Reinforcement = 8Nos.10 mm nominal diameter Ties 8mm diameter @ 50mm spacing Ties clear cover = 15mm	
Model 2	0 30 60 90	Longitudinal Reinforcement = 6Nos.12 mm nominal diameter Ties 8mm diameter @ 50mm spacing Ties clear cover = 15mm	
Model 3	0 30 60 90	Longitudinal Reinforcement = 4Nos.16 mm nominal diameter Ties 8mm diameter @ 50mm spacing Ties clear cover = 15mm	

3. Finite Elements in Ansys

The following are the finite elements used for the analysis.

3.1 Types of element

3.1a Reinforced concrete

Solid 65, eight node solid element is used to model the concrete it has eight nodes with three degrees of freedom at each node include translations in the nodal x, y and z directions the element is capable of predicting of plastic deformation, cracking in three orthogonal directions and crushing.

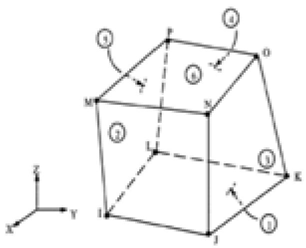


Figure 1: Solid 65-3-D solid

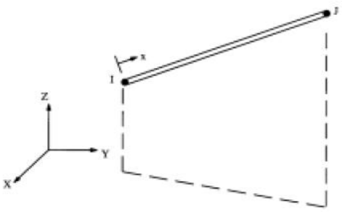


Figure 2: Link8 element

Steel reinforcement is modelled using link 8 element. Two nodes are required for this element and each node has three degrees of freedom include translations in the nodal x, y, and z directions. The element is also capable of plastic deformation.

3.2.b Steel Plates

Solid 65, eight node solid element is used to model the steel plates at the column supports. The element is defined with eight node having three degrees of freedom at each node include translations in the nodal x,y and z directions

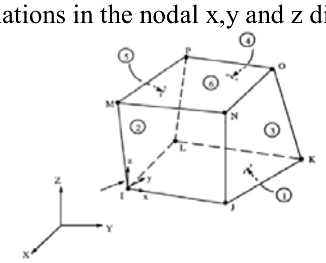


Figure 3: Solid45-3-D solid

4. Results and Discussions

From the analytical study, it shows that all the three models shows a definite variation in the values for ultimate load, load for first crack, and maximum deflection under same concentric and eccentric compression.

All the three model subject to same concentric loading, model 3 carries more ultimate load than the other two models.As the eccentric distance increases, the load carrying capacity decreases for all the three models. For small eccentricity (ie,30mm), all the three models carries almost equal ultimate load. As the eccentricity increases there exhibit a clear variation in the value for ultimate load. For 60mm eccentric case, model 3 carries more ultimate load and for 90mm eccentricity, model 2 carries more load than the other two models.

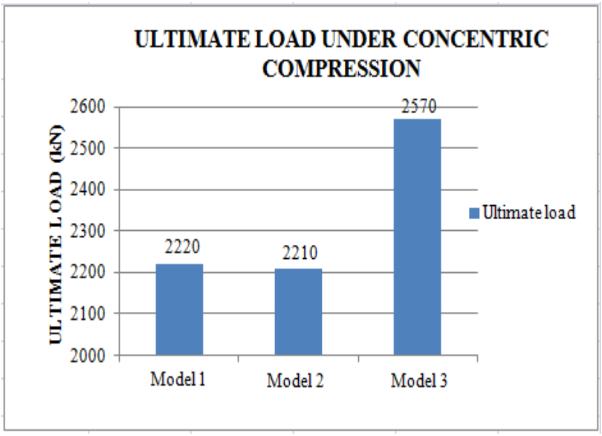


Chart 1: Graphical representation of ultimate load under concentric compression for three models.

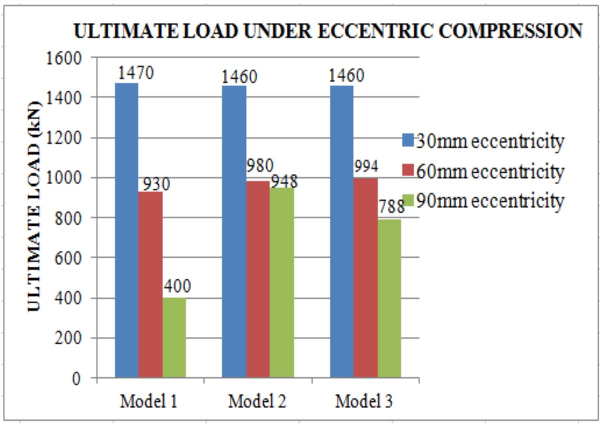


Chart 2: Graphical representation of ultimate load under eccentric compression for three models

Table 2: Results for Model 1

Eccenticity (mm)	Maximum Deformation (mm)	Ultimate Load (kN)	First crack load (kN)
0	3.873	2220	1900
30	8.28	1470	1260
60	11.27	930	270
90	10.62	400	124

Table 3: Results for Model 2

Eccentricity (mm)	Maximum deformation(mm)	Ultimate load (kN)	First crack load (kN)
0	7.59	2210	1910
30	7.77	1460	1250
60	9.21	980	640
90	11.45	948	270

Table 4: Results for Model 3

Eccentricity (mm)	Maximum deformation(mm)	Ultimate load (kN)	First crack load (kN)
0	11.23	2570	1980
30	7.52	1460	1300
60	11.80	994	280
90	39.18	788	130

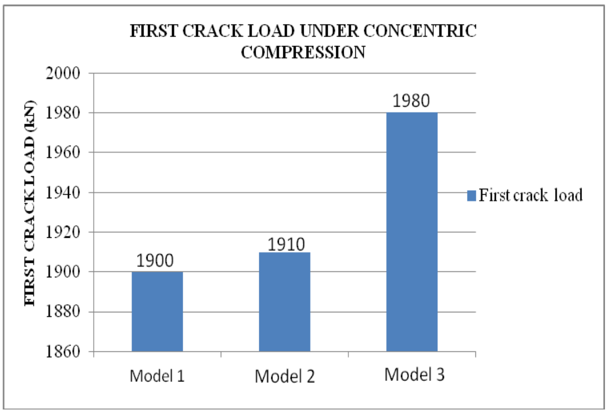


Chart 3: Graphical representation of first crack load under concentric compression for three models.

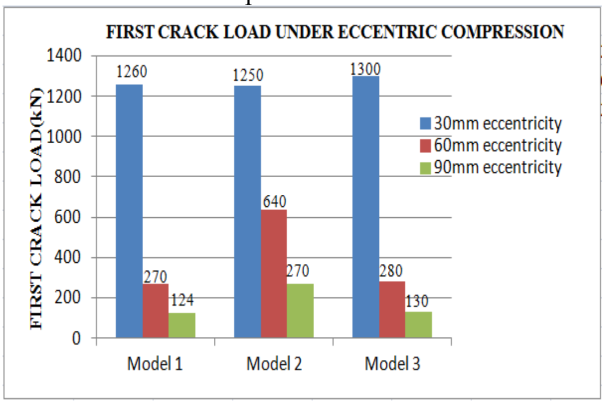


Chart 4: Graphical representation of first crack load under eccentric compression for three models.

Model 3 posses a high load value for first crack. Its about 1980kN. While the other two model exhibits an almost nearer load value for first crack. It means that the first crack occur first in model 1 and model 2 and then only it occurs in model 3 which means model 3 behaves stable than other two under concentric loading. It infers that for small eccentricity, all the three models posses almost nearer load value for first crack. But as the eccentricity increases model 2 behaves more stable than the other two.

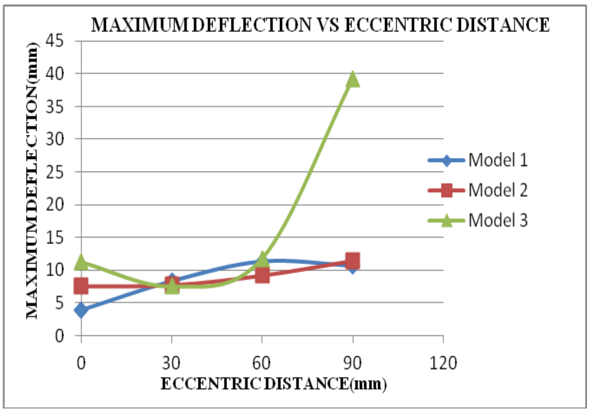


Chart 5: Graphical representation of maximum deflection with eccentricity for three models.

There is a definite increase in the maximum deformation value according to the increase in eccentric distance for model 2. For model 3 the variation is irregular. For large eccentricity, model 3 posses a very high deformation value than the other two models. It is observed that model 1 and model 2 exhibit a moreover similar pattern than the model 3. All the three model subject to same concentric loading, model 3 carries more ultimate load than the other two models.

5. Conclusion

The selection of suitable reinforcement both transverse and lateral reinforcement plays an important role in the behaviour of concrete structures. In addition to the selection the selection of reinforcement, the arrangement also important. From the current study following conclusions were made.

1) Ultimate load capacity of the column differs for all the three cases. Maximum ultimate load under concentric compression is obtained for model 3.
2) The first and second model have a gradual increase in the maximum deformation value according to the eccentric distance. While the model 3 behaves irregularly with the eccentric distance.
3) For the small eccentricity, model 1 behaves more stable than other two models.
4) As the eccentricity increases, the model2 behaves more stable than other two.
5) It shows that small amount of confinement is required for small concentric load.
6) As the amount of confinement increases, the value for maximum deformation decreases under concentric compression.
7) Since the model 3 posses small amount of confinement, it prone to first crack under eccentric compression very earlier than the other two models.

6. Acknowledgement

I would like to express my sincere gratitude to God Almighty who has give me the strength to complete this work. Also i have been fortunate in having Mrs. Indu Susan Raj (Assistant Professor, Department of Civil Engineering, IIET),as my guide, whose timely guidance, advices and inspiration helped me in the preparation and presentation of this work in time.
I would like to extent my special thanks to my parents and friends.

References

[1] Ibrahim S.I ''*Non Linear Finite Element Analysis of Confined HSC Under Concentric and Eccentric Loadings"*, Journal of Engineering and Development, Vol. 16, No.3,2012.

[2] Edward G. Nawy, P.E., "*Prestressed Concrete A Fundamental Approach*", Third Edition, New Jersey 07458, 2006.

[3] Teng-Hooi Tan and Ngoc-Ba Ngugen," *Flexural Behavior of High-Strength Concrete Columns"*, ACI Structural Journal, V.102, No. 2, March-April 2005.

[4] Wolanski A.J,*"Flexural Behavior of Reinforced and Prestressed Concrete Beams using Finite Element Analysis"* , MSc. Thesis, Faculty of the Graduate School, Marquette University, Milwaukee, Wisconsin, May 2004.

[5] Assa,B., Nishiyama, M., and Watanabe, F., *" New Approach for Modeling Confined Concrete Rectangular Columns"*. Journal of Structural Engineering, ASCE, V. 127, No.7, July 2001.

[6] Junichi Sakai And Kazuhiko Kawashima, "*An Unloading and Reloading Stress-Strain Model for Concrete Confined by Tie Reinforcements"*. International Journal for Civil Engineering, Vol. 1, No. 2, pp. 151–164, 2000.

[7] ACI 318-89, 1992,"*Building Code Requirements for Reinforced Concrete*," A CI Manual of Concrete Practice Part 3-1992, American Concrete Institute.

Author Profile

Arathy Nath P, M.Tech student of IIET,Nellikuzhi specializing in Structural Engineering and Construction. Management.

Indu Susan Raj, Assistant Professor of IIET, Nellikuzhi specialized in Structural Engineering and Construction Management.